PROJECT NAME

AURORA

aion DIY GUITAR EFFECTS

BASED ON
Ross Compressor / MXR® Dyna Comp®

BUILD DIFFICULTY
Intermediate

EFFECT TYPE
Compressor / Sustainer

DOCUMENT VERSION
1.0.1 (2024-08-08)

PROJECT SUMMARY
The original guitar compressor, still a favorite of guitarists after over 40 years. The Aurora makes several improvements to the original circuit as well as adding a few new features.

IMPORTANT NOTE

This documentation is for the **kit** version of the project. If you purchased the PCB by itself, please use the PCB-only version of the documentation instead. The circuit is the same, but the instructions are completely different due to the specialized parts and assembly methods used in the kit.

TABLE OF CONTENTS

INTRODUCTION

If this is your first pedal, welcome to the hobby and thank you for choosing Aion FX. You've just joined a community of over 100,000 people around the world with a passion for building homemade noise machines using obsolete electronics technology, and we're glad to have you!

If you've done this before, it's great to see you again and we're confident you'll find this build experience an enjoyable one.

Aion FX kits are designed to empower anyone to build a high-quality pedal, no matter the skill level. The pedalbuilding hobby has traditionally had a steep learning curve, but don't be overwhelmed—we've done all the hard work for you. All you need to do is follow these instructions and you'll be on your way to transforming your tone.

There are a few things to go over before you get started.

- **You're going to have to get your hands dirty**—there's no way around it. Nothing here comes preassembled, and you'll have to learn the skills to put it all together. This document will walk you through everything you need, but be prepared to learn a few things along the way.
- **This will take time**. Plan on about two hours start to finish. It may take even longer if it's your first time building. Don't rush it. If you find yourself getting frustrated or overwhelmed, take a break and come back in a couple of hours or the next day.
- **No direct technical support is offered.** There are several DIY forums and Facebook groups with thousands of members who enjoy troubleshooting and teaching. But please be sensitive to the fact that the staff at Aion FX is minimal, and every minute spent helping individuals in private is time that can't be spent on new project development.
- **There is no implied guarantee of a final product.** Aion FX provides the ingredients and the recipe, but you are responsible for putting everything together to make it work. We've tried to make the process as clear and accessible as possible, but it must be expressly stated that purchasing the kit is not a guarantee that you will end up with a working pedal.

It's recommended to read through all of the instructions before you start, particularly if you've never built a pedal before. If you familiarize yourself with the entire process ahead of time and you know what the goal looks like, each step will make more sense.

Now, on to the fun stuff!

PACKING LIST

This is a list of all the parts that are included with the kit, grouped by value. For a list of all the parts based on their PCB part numbers, please see page 26.

If you find that any parts are missing or damaged, please fill out the Missing Parts form.

Film Capacitors

NAME	QTY
1n	1
2n2	1
10n	4
100n (0.1 or "µ1J100")	1
1uF	1

Electrolytic Capacitors

NAME	QTY
1uF	4
10uF	1
47uF	1
100uF	1

MLCC Capacitors

NAME	QTY
150pF	1
180pF	1
330pF	1
820pF	1
100n (marked "104")	1

IC & Sockets

NAME	QTY
LM13700 or NE5517	1
8-pin socket	2

Resistors

NAME	QTY
100R	1
1k	3
10k	10
15k	1
27k	2
56k	1
150k	1
220k	2
390k	1
470k	2
1M	4
2M2	1

Diodes

NAME	QTY
1N5817	1
1N914	2

Transistors

NAME	QTY
2N5088	6

PACKING LIST (CONT.)

Potentiometers

NAME	QTY
50kA	1
250kC	1
500kC	1
Dust cover	3
Knob	3
Mounting nut, potentiometer, 0.44"	3
Lock washer, potentiometer, 0.5"	3
Outer washer, potentiometer, 0.475"	3

Switches

NAME	QTY
Toggle switch, SPDT on-off-on	1
Mounting nut, toggle switch, 0.36"	1
Lock washer, toggle switch, 0.4"	1
Dress nut, toggle switch, 0.375"	1
Stomp switch, 3PDT	1
Mounting nut, stomp switch, 0.6"	2
Lock washer, stomp switch, 0.6"	1
Dress nut, stomp switch, 0.77"	1

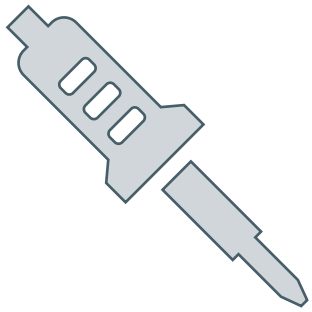
Other

NAME	QTY
LED bezel	1
LED, blue	1
9V battery snap	1
DC jack	1
Input/output jack	2
Mounting nut, jack, 0.54"	4
Outer washer, jack, 0.6"	2
Lock washer, jack, 0.5" (thin)	2
Enclosure	1
Enclosure screws	4
PCB, main circuit	1
PCB, footswitch	1
PCB, input/output/DC	1

Wiring

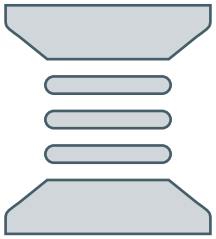
NAME	QTY
3-strand wire assembly, 70mm	2
4-strand wire assembly, 108mm	1
3-pin wire assembly header	2
4-pin wire assembly header	1

TOOLS NEEDED



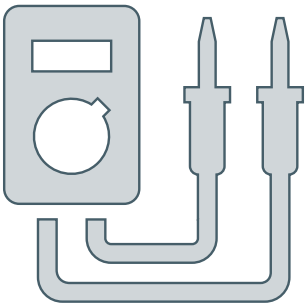
SOLDERING IRON

Temperature-adjustable is recommended. The optimum soldering temperature is 700-725° F (371-385° C) for leaded solder, or 750° F (400° C) for lead-free.



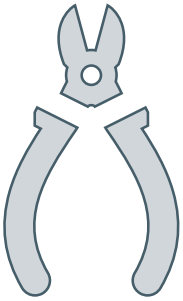
SOLDER

Preferably 63/37 or 60/40 leaded solder. Lead-free is more difficult to use, so if that's the only type you can get, it's best to watch tutorials that are specific to lead-free solder.



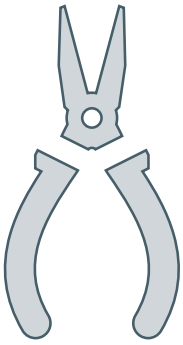
DIGITAL MULTIMETER (DMM)

Most cheap ones in the $10-30 range are fine for what we're doing. Make sure it has audible continuity testing (i.e. it beeps at the lowest resistance) and transistor hFE measurement.



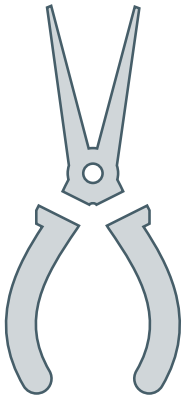
WIRE SNIPPERS

Also called nippers or wire cutters. The Hakko CHP-170 is the best you can get for less than $10.



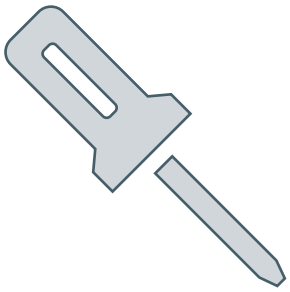
FLAT-NOSE PLIERS

Many general-purpose uses, but particularly tightening the nuts of pots, switches and jacks. Quicker than changing out sockets on a ratchet.



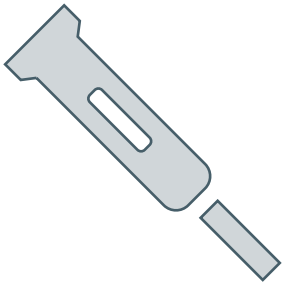
NEEDLE-NOSE PLIERS

These are used for bending leads on components and other general uses. Use the smaller type with a tip that's approximately 0.05" (1.25mm) wide.



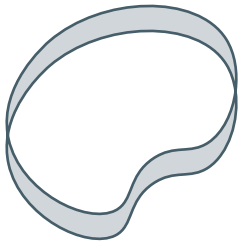
SCREWDRIVER (PHILLIPS)

Used for the enclosure screws. Get a powered driver if you'll be building a lot of pedals!



FLAT SCREWDRIVER (SMALL)

This is used for tightening the set screws on the knobs. The tip should be no more than 0.1" (2.5mm) wide.



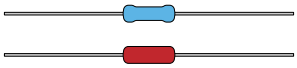
RUBBER BAND

Yes, a plain old rubber band. This is used to tighten the dress nut to avoid scratching or denting it (which can happen with metal tools).

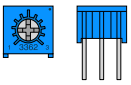
COMPONENT IDENTIFICATION

If you've never built a pedal before, you'll need to know what all the components are. These are shown actual size. (Not all of these types of components may be part of this kit.)

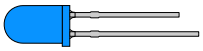
RESISTOR



TRIM POTENTIOMETER

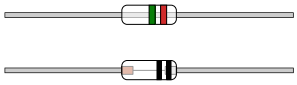


LED



SILICON DIODE

GERMANIUM DIODE



These are very fragile, so be careful when handling them.

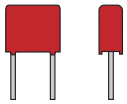
RECTIFIER DIODE

Some Schottky diodes also look like this.

SCHOTTKY DIODE

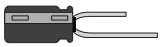
ZENER DIODE

FILM CAPACITOR



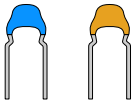
Not polarized. Color may vary by brand and type.

ELECTROLYTIC CAPACITOR



Polarized. The negative side is marked.

MLCC

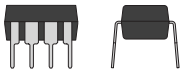


Not polarized. MLCC stands for "multi-layer ceramic capacitor."

TANTALUM CAPACITOR

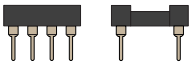
Polarized. The positive side is marked.

OP-AMP / IC

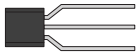


Charge pumps and delay chips also look like this. They may have more than 8 legs.

IC SOCKET



TRANSISTOR OR JFET



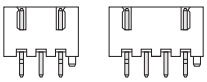
Some voltage regulators also look like this.

SNAP-APART SOCKET

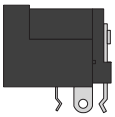
It's recommended to use a razor blade to separate these cleanly.

WIRE ASSEMBLY

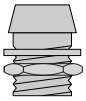
WIRE ASSEMBLY HEADER



DC JACK



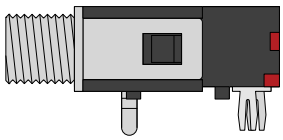
LED BEZEL



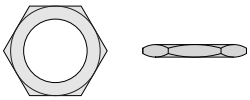
HARDWARE IDENTIFICATION

The hardware comes unassembled, so you'll need to sort & identify each of the pieces. The diagrams below are actual size, so you can set them against the printed page to identify them if needed.

I/O JACK

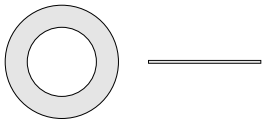


MOUNTING NUT



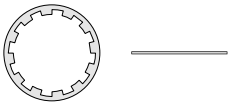
DIAMETER: 0.54" / 13.7mm

OUTER WASHER



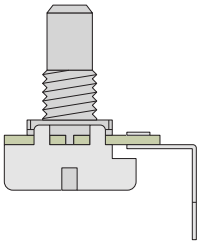
DIAMETER: 0.6" / 15.2mm

LOCK WASHER

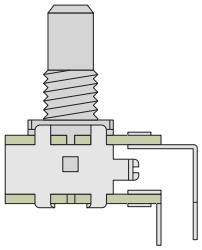


DIAMETER: 0.5" / 12.7mm

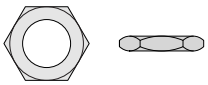
POTENTIOMETER (SINGLE)



POTENTIOMETER (DUAL)

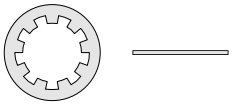


MOUNTING NUT



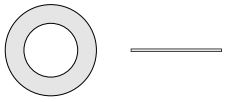
DIAMETER: 0.44" / 11.2mm

LOCK WASHER



DIAMETER: 0.5" / 12.7mm

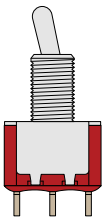
OUTER WASHER



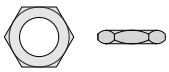
DIAMETER: 0.475" / 12mm

KNOB

TOGGLE SWITCH

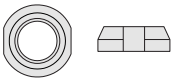


MOUNTING NUT



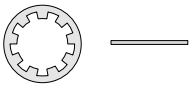
DIAMETER: 0.36" / 9.1mm

DRESS NUT



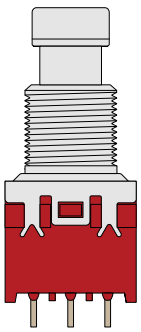
DIAMETER: 0.375" / 9.5mm

LOCK WASHER

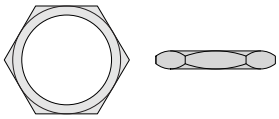


DIAMETER: 0.4" / 10.1mm

FOOTSWITCH

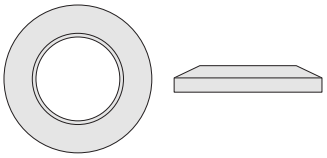


MOUNTING NUT



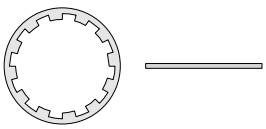
DIAMETER: 0.6" / 15.2mm

DRESS NUT



DIAMETER: 0.77" / 19.6mm

LOCK WASHER

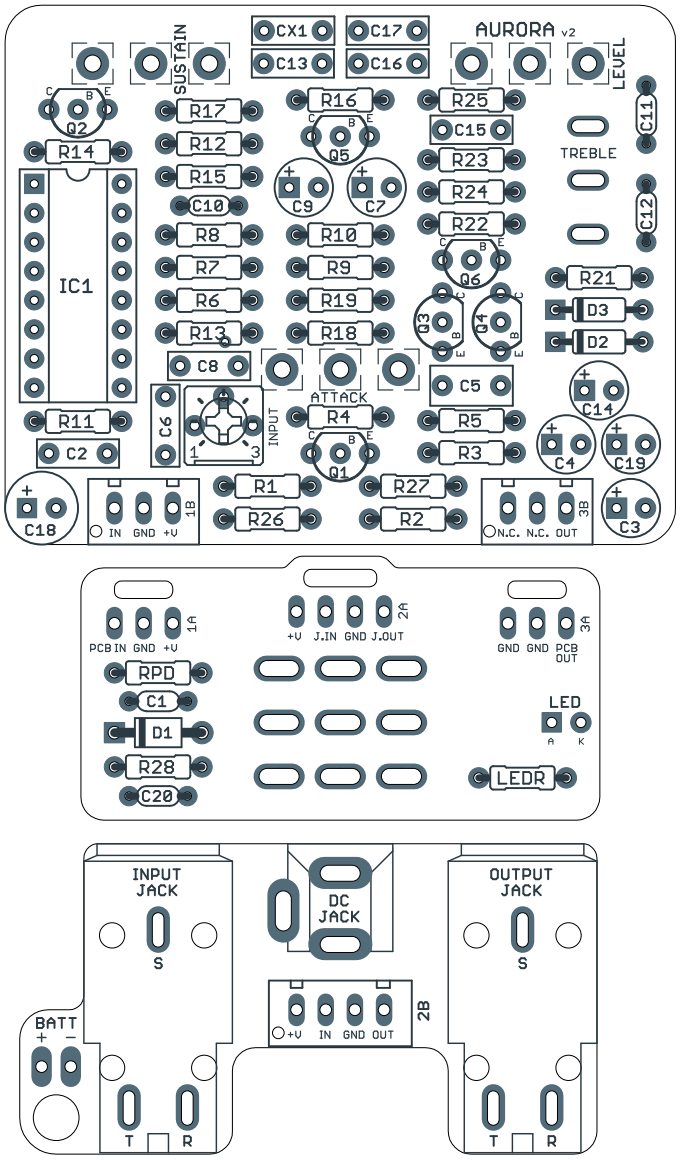


DIAMETER: 0.6" / 15.2mm

PCB ASSEMBLY OVERVIEW

Now it's time to start building!

The first thing you need to do is separate the PCBs into 3 separate boards and break off the tabs from each using needle-nose or flat-head pliers. You should be left with the PCBs shown to the right.

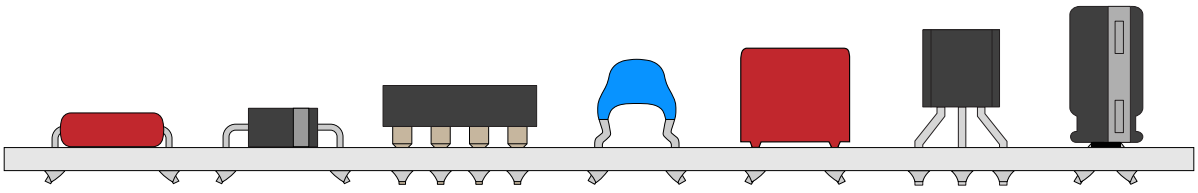


The general principle for PCB population is that you want to work in layers from shortest components (i.e. lowest-profile) to tallest. This way, when you turn the PCB upside down, the components are held in place when soldering.

Generally speaking, you should populate the components in this order:

1. Resistors
2. Diodes
3. IC sockets
4. MLCC capacitors
5. Film capacitors
6. Transistors
7. Electrolytic capacitors

Not all of these component types are included in each kit, so skip them if they aren't applicable. Some types of film capacitors are taller than electrolytics, so those can be done last.



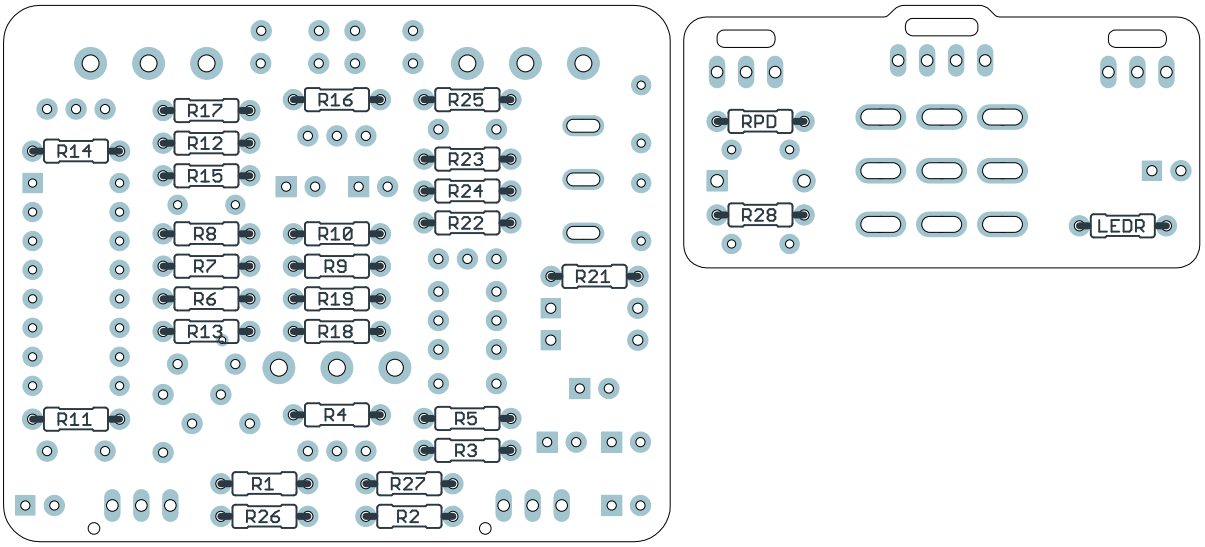
RESISTORS

PART	VALUE
R1	10k
R2	470k
R3	470k
R4	10k
R5	10k
R6	1M
R7	1k
R8	1k

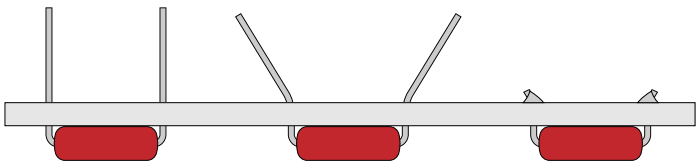
PART	VALUE
R9	220k
R10	220k
R11	1M
R12	15k
R13	150k
R14	27k
R15	10k
R16	10k

PART	VALUE
R17	1M
R18	10k
R19	390k
R21	1M
R22	1k
R23	10k
R24	10k
R25	10k

PART	VALUE
R26	56k
R27	27k
R28	100R
RPD	2M2
LEDR	10k



Using the parts list above, populate the resistors by pushing them through the holes and bending the leads outward at an angle to hold them in place. Resistors are not polarized, so they will work in any direction. Turn the board upside-down to keep the components held in place while you solder.

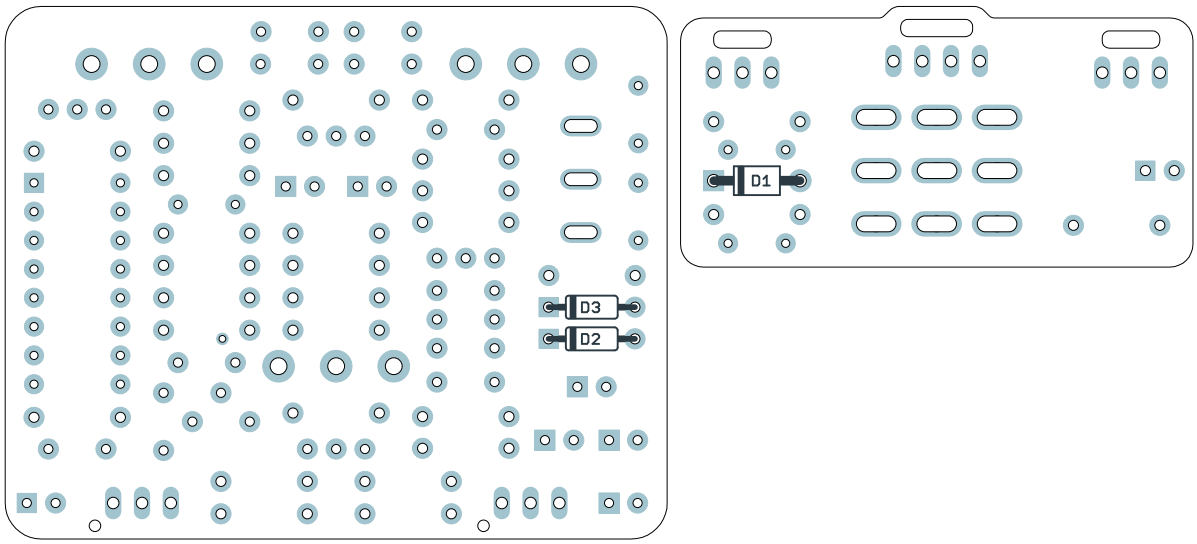


You'll use this same technique for most of the other components as well.

If this is your first time soldering, watch tutorial videos on YouTube and make sure you get it down before you begin. You don't want to practice or experiment on this board!

DIODES

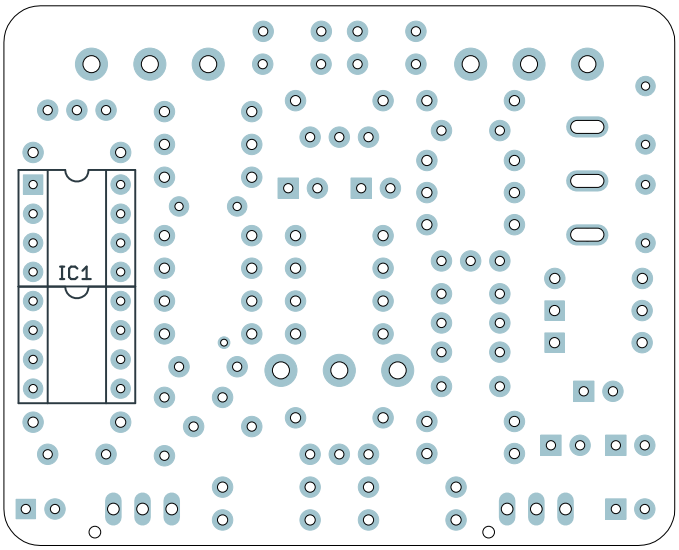
PART	VALUE
D1	1N5817
D2	1N914
D3	1N914



Next, you'll populate the diodes. Diodes are polarized, so make sure to identify the polarity band (which indicates the "cathode", or negative side) and match the band to the footprint on the PCB.

IC & SOCKETS

PART	VALUE
IC1	LM13700 or NE5517



Next up are the IC sockets. The Aurora uses one 16-pin IC, but there are two 8-pin sockets. Install them as shown in the diagram above.

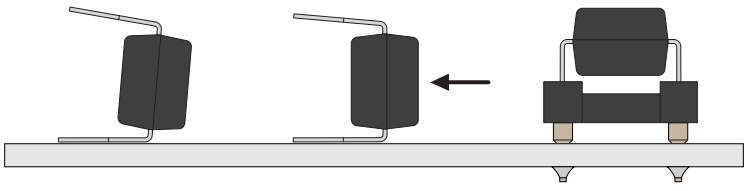
You can't bend the leads of a socket like you can with the other components, so it won't stay in on its own until it is soldered. Again, it's much easier to do this upside down with gravity holding it in place for you, so you'll want do this before you do any of the taller components.

Installing the IC

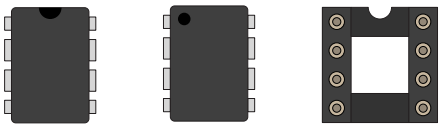
Don't insert the IC into the sockets just yet. We will do this in a later step, after we've finished soldering the tallest components (the polarized capacitors). This information is just listed here for reference.

The legs of the IC are bent outward slightly during manufacturing, so they'll need to be bent back inward before it can be inserted into the socket.

It's easiest to do this by laying the IC legs against the table and bending the body itself so all eight legs on the side are straightened out at once. Then, flip it and do the other side.



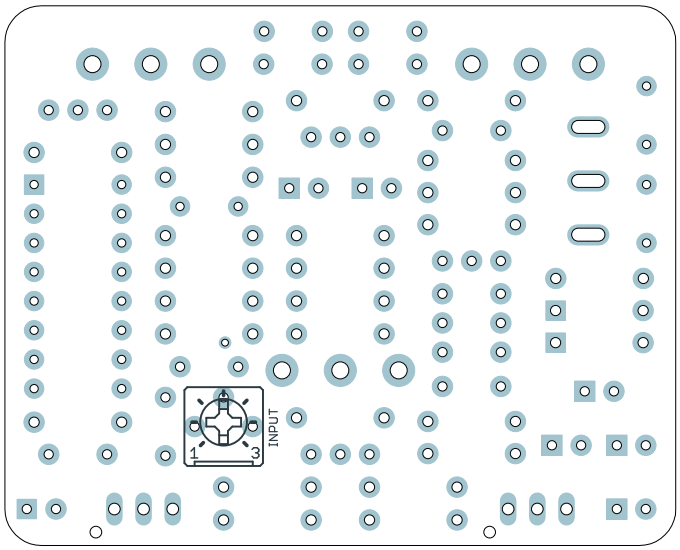
ICs may have two different orientation marks: either a dot in the upper-left or a half-circle notch in the middle of the top side. Some ICs have both marks. This shows which way the IC should be rotated when inserting it into a socket. The socket also has a half-circle notch to indicate the orientation.



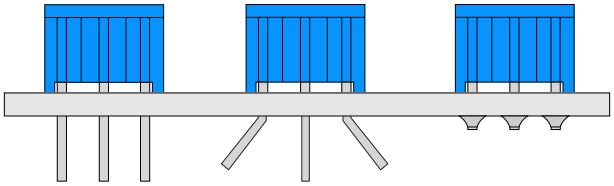
DIP-8 ICs shown for reference

TRIMMER

PART	VALUE
INPUT	100k trimmer



The input trimmer comes next. It can be soldered like a normal component, by bending the legs outward as shown:

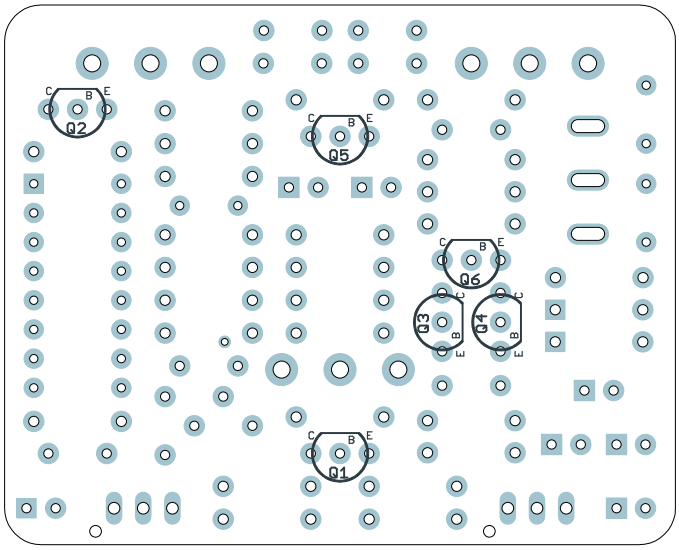


This trimmer can be used to attenuate the input signal so the compressor works better with high-output instruments such as active pickups or keyboards.

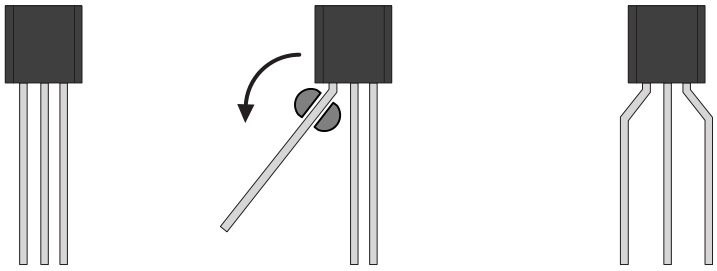
The default position is all the way down. If you experience any clipping, turn it up until the clipping goes away. This gradually reduces the input signal in order to keep from overloading the compressor IC.

TRANSISTORS

PART	VALUE
Q1	2N5088
Q2	2N5088
Q3	2N5088
Q4	2N5088
Q5	2N5088
Q6	2N5088

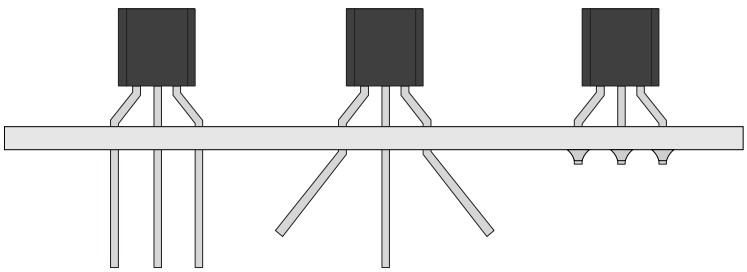


Now we'll do the transistors. If the legs are not already bent into 0.1" spacing, use your needle-nose pliers to bend the outer two legs as shown.



Since these are just used as buffers and don't need to be selected for gain, sockets are not necessary. You can just solder them directly to the board.

Bend the outer leads to hold it in place on the board. Then, solder them and clip the leads.

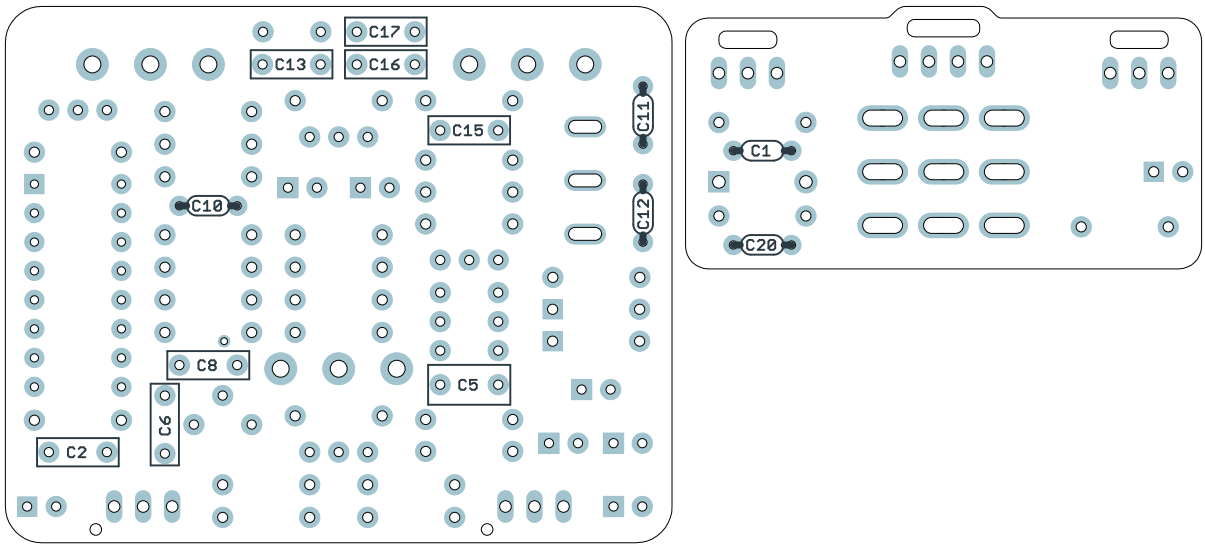


CAPACITORS (NON-POLARIZED)

PART	VALUE
C1	150pF MLCC
C2	10n film
C5	1uF film
C6	2n2 film
C8	10n film

PART	VALUE
C10	180pF MLCC
C11	330pF MLCC
C12	820pF MLCC
C13	10n film
C15	10n film

PART	VALUE
C16	100n film
C17	1n film
C20	100n MLCC
CX1	OMIT



After the sockets come the box film and MLCC capacitors. These are all several different heights, but there aren't as many, so just do them all at once. Bend the leads at an angle to hold them in place.

MLCCs and box capacitors are not polarized and will work in any direction. To keep things neat, though, it's recommended to put them all facing the same way.

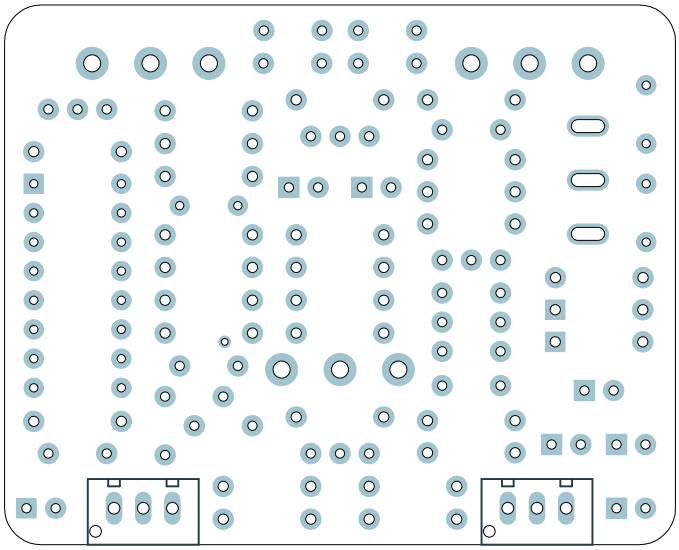
Note: Depending on the type, the box film capacitors may have their value printed on either the top or the side. Usually the red ones have it printed on the side while the blue or gray ones have it on the top.

C1, C10, C11, and C12 are blue MLCCs taped to cardboard. The value is written on the cardboard.

C20 (100n MLCC) is always yellow. It can be hard to read the code since it's so small, so it's easier to identify this one by color.

CX1 is not used in this kit and should be left empty on the PCB.

WIRE HEADERS

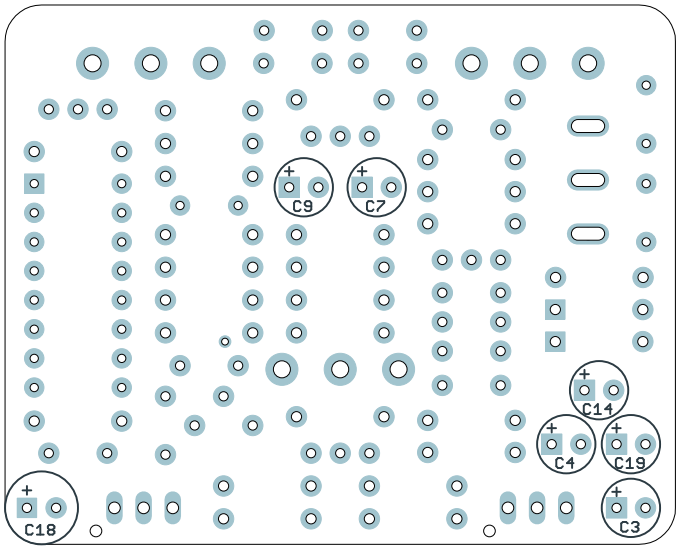


Install the two 3-pin headers (wire connectors) as shown above. These have a polarity pin, so as long as they are pressed all the way down, there's only one possible way to install them. They do fit pretty tightly in the holes, though, so press firmly.

There's also a 4-pin header on the I/O board that we will do in a later step.

CAPACITORS (POLARIZED)

PART	VALUE
C3	1uF electro
C4	1uF electro
C7	1uF electro
C9	1uF electro
C14	10uF electro
C18	100uF electro
C19	47uF electro

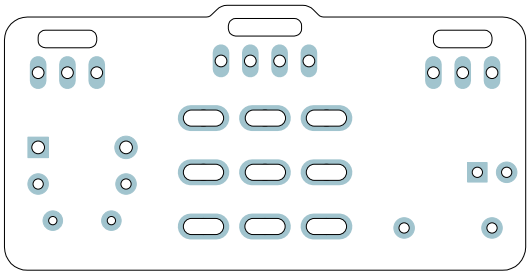


Populate the electrolytic capacitors. These are the tallest components so we save them for last. They are polarized (i.e. they will only work in one direction), so note the vertical mark that indicates the negative side. The longer leg is positive and fits in the square pad.

These are the last of the on-board components. Now is the time to go back to page 12 and insert the IC into the socket.

FOOTSWITCH PCB

PARTS
3-strand wire assembly (2)
4-strand wire assembly



Next, it's time to finish up the footswitch board. You should have done most of the on-board components on this board in a previous step, but if not, go back and do those.

There will be one longer assembly with 4 wires and two shorter ones with 3 wires. The longer one goes in the middle and the shorter ones go on the left and right sides. The wire assemblies should then be soldered to the footswitch board as shown.

STEP 1

First, thread the wire through the strain-relief slots, with the blue side facing outward and the PCB's previously-installed components facing up.

For now, pull it through as far as it can go.

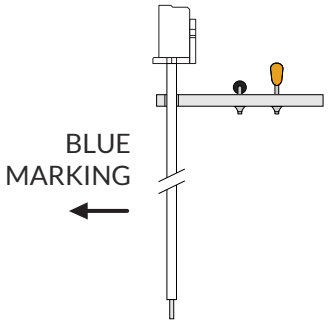
STEP 2

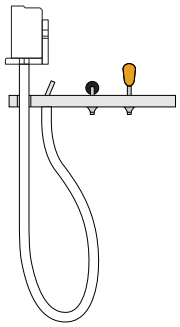
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

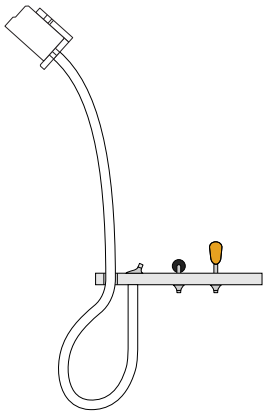
On the top side of the PCB, bend the exposed wires backward so it holds the wire in place. Pull the header back up through the slot partway.

STEP 3

Then, solder the wires from the top. This is the trickiest part of the whole build. You want to solder the pads without touching the iron to the wires themselves and risking burning through the insulation. It helps to use a sharp or narrow tip on the soldering iron.



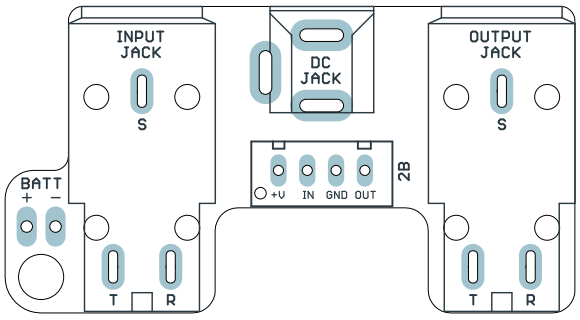




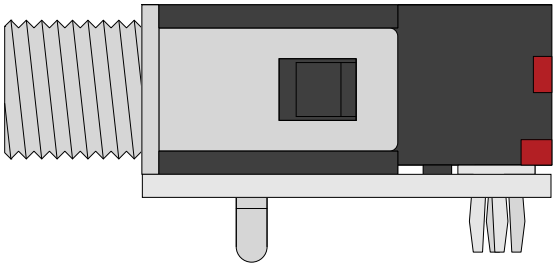
Once all three wire assemblies are soldered, set the footswitch PCB aside. We'll solder the actual footswitch and LED in a later step.

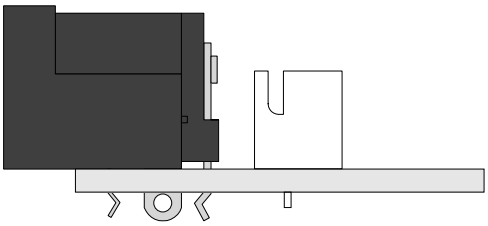
INPUT/OUTPUT PCB

PARTS
Input & output jacks
DC jack
Wire header
9V battery snap



Almost done! Get the two input/output jacks, the DC jack and the wire header and snap them in place. The PCB is designed for them to fit securely, so you can do them all at once before flipping and soldering.



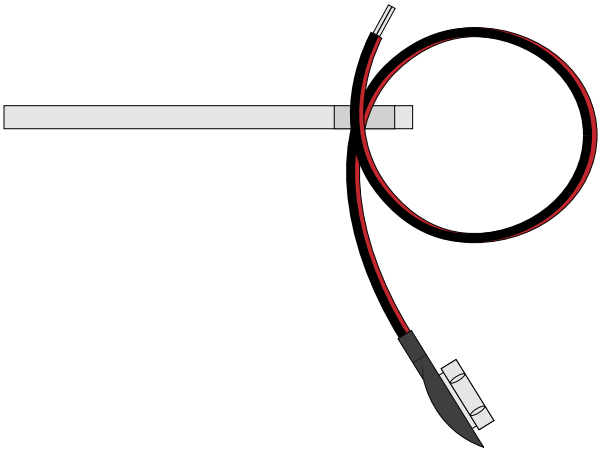


After you've soldered everything, make sure to **snip the leads on the I/O jacks as close as possible to the PCB**. There's not a lot of clearance between the bottom of this board and the top of the main PCB once everything is in place, and you don't want the pins to short against anything on accident.

Next, we'll hook up the 9V battery connector. **This is optional.** Not everyone uses batteries. But, if you do, this pedal should last a long time on a single 9V so you won't need to change it very often.

STEP 1

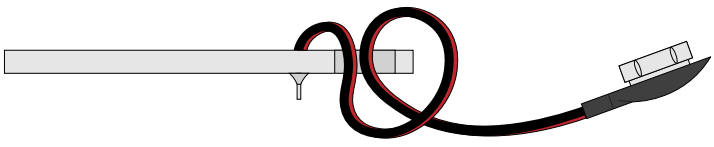
Thread the battery snap leads through the strain-relief hole twice so it forms a single loop.



STEP 2

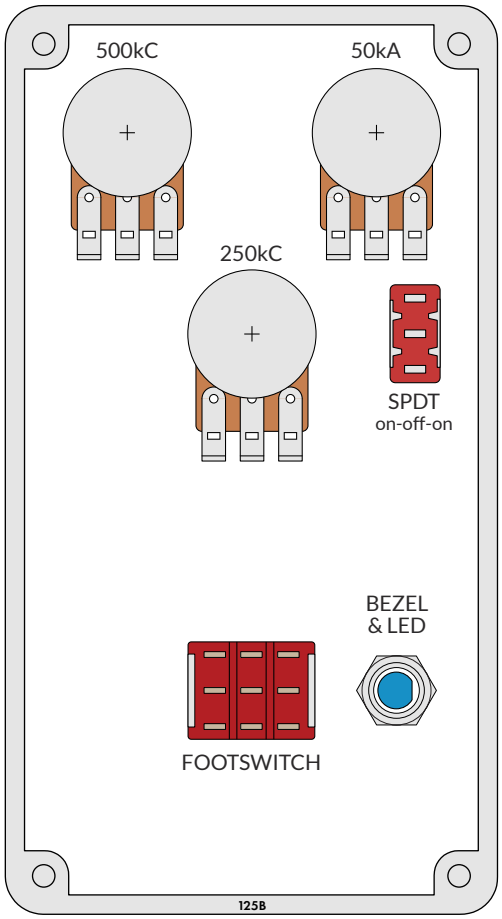
Bend the exposed wires back down and solder them into the pads. Red is positive (+), black is negative (-). After soldering, pull it tight.

For even more strain relief, you can thread the snap through the loop to form a knot. (not shown)

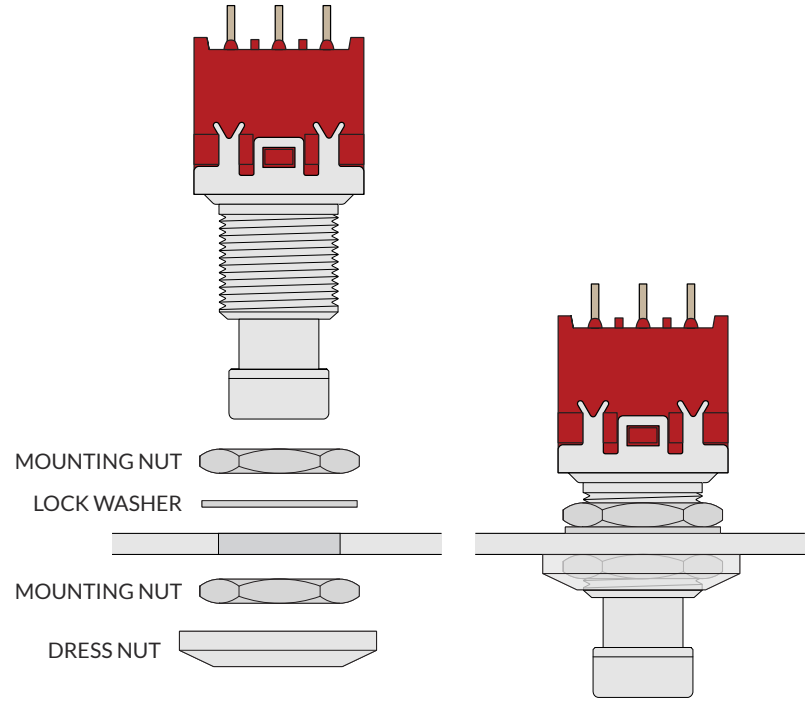


ENCLOSURE LAYOUT: PANEL MOUNTS

Attach the hardware to the enclosure as shown. (The I/O board is done in a later step.)

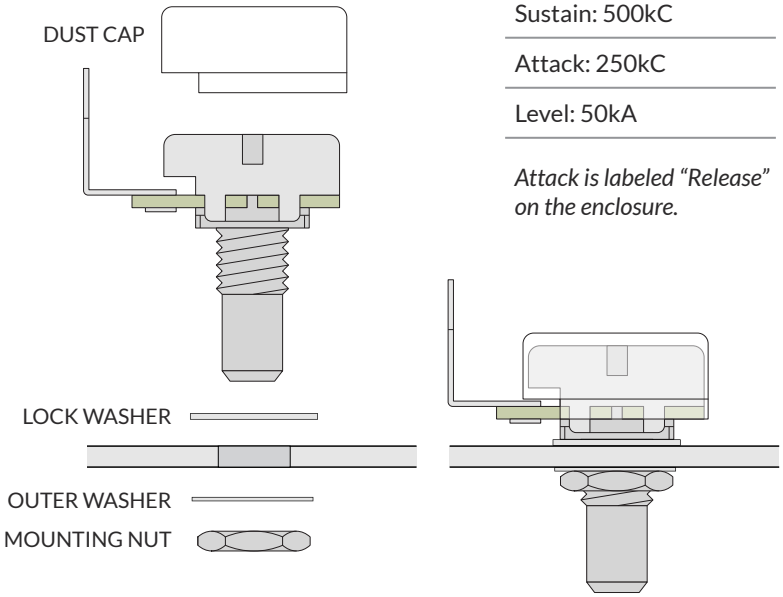


FOOTSWITCH



The dress nut fits over the top of the mounting nut and is for aesthetic purposes only. Wrap a rubber band around it to use as a grip when tightening. Do not use metal tools on it or you run the risk of scratching or denting it.

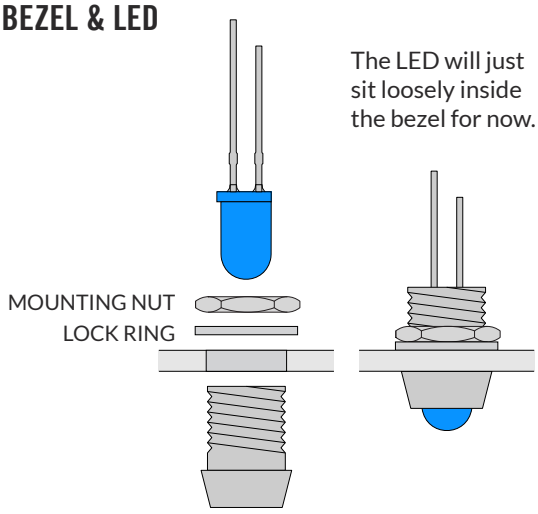
POTENTIOMETERS



Sustain: 500kC
Attack: 250kC
Level: 50kA

Attack is labeled "Release" on the enclosure.

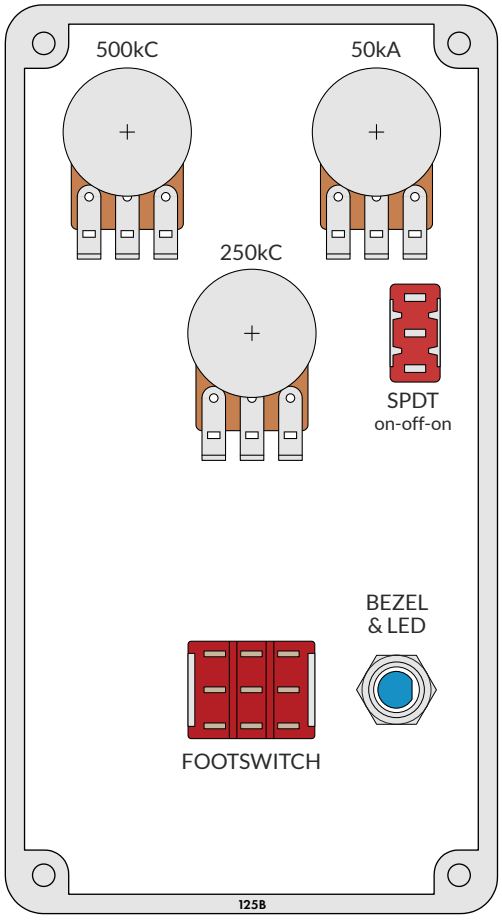
BEZEL & LED



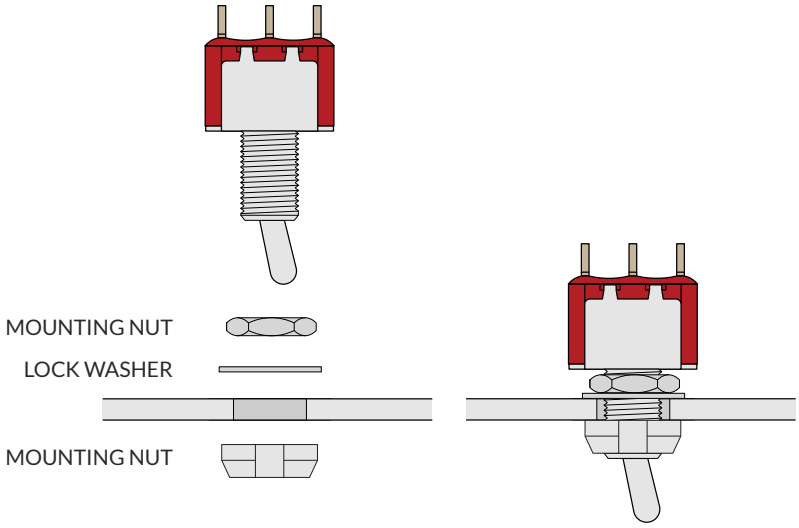
The LED will just sit loosely inside the bezel for now.

You'll need to hold the bezel in place when tightening the nut. The top of the bezel is fairly sharp, so try using a rubber band for grip instead of pressing your finger against the bottom.

ENCLOSURE LAYOUT: PANEL MOUNTS (CONT.)

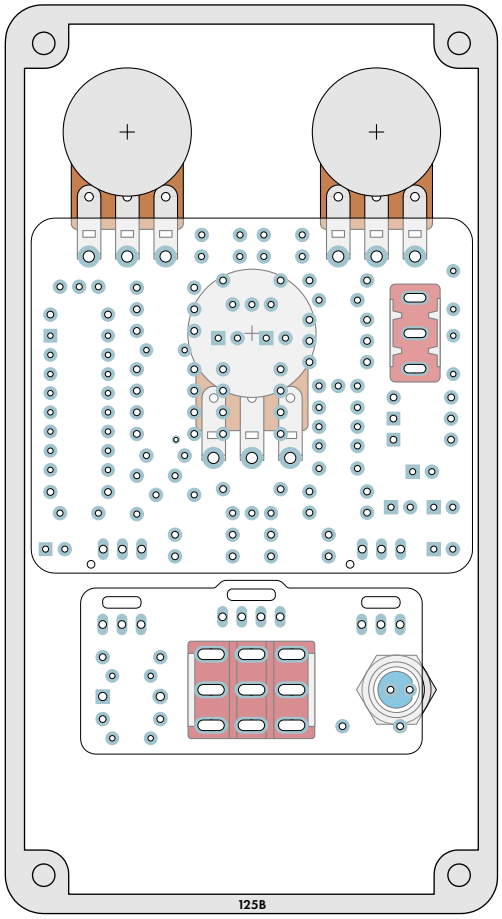


TOGGLE SWITCH



The dress nut acts as a mounting nut, unlike the footswitch dress nut. Use flat-nose pliers on the flat sides of the nut to tighten securely.

ENCLOSURE LAYOUT: MAIN & FOOTSWITCH PCBS



After all the components are affixed to the enclosure as shown on the previous page, place the main PCB on top of the potentiometers and toggle switches as in the diagram to the left.

You may need to adjust the position of the potentiometers and toggles slightly if they are not aligned straight.

Once all of the pins are through their holes and the PCB is laying flat, solder each of the pins from the top. Be careful not to touch any of the surrounding components with the soldering iron.

After you've finished soldering the pots, **clip the leads as close as you can to the main PCB**. This is more important with the two uppermost pots because the input/output PCB overlaps them and you need to avoid any of the components shorting. (The toggle switch lugs do not need to be clipped.)

Next, move to the footswitch board and solder the 3PDT switch.

The LED is last. Before soldering the LED, double-check to **make sure the flat side of the LED is facing to the right**, as shown in the diagram, and that the short leg is coming through the pad on the right. It won't work if it's turned the other way. Then, clip the leads of the LED.

Why solder everything inside the enclosure before testing it?

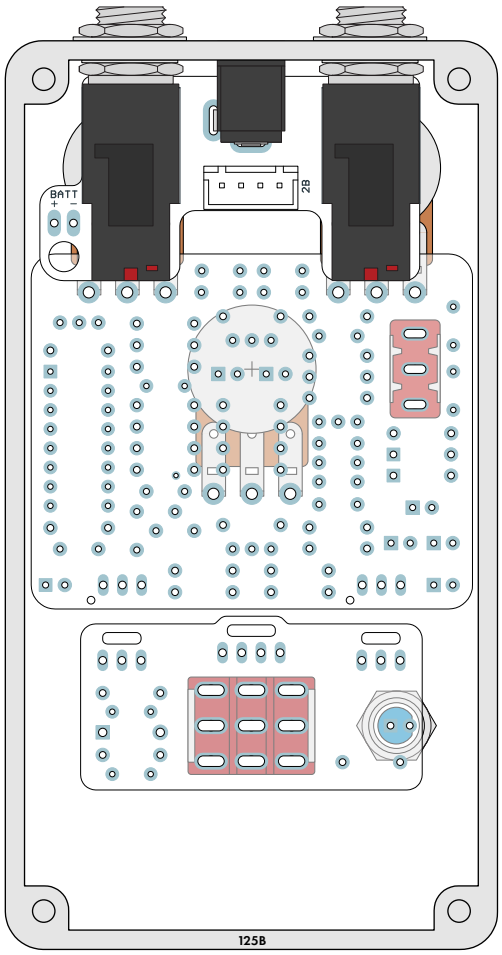
"Rock it before you box it" is conventional wisdom in pedalbuilding, and you'll often hear it recommended that builders should test the circuit before putting everything inside the enclosure. However, Aion FX projects are designed to be extremely easy to remove from the enclosure for troubleshooting, with no desoldering required—so with these kits, it's actually much easier to "box it before you rock it".

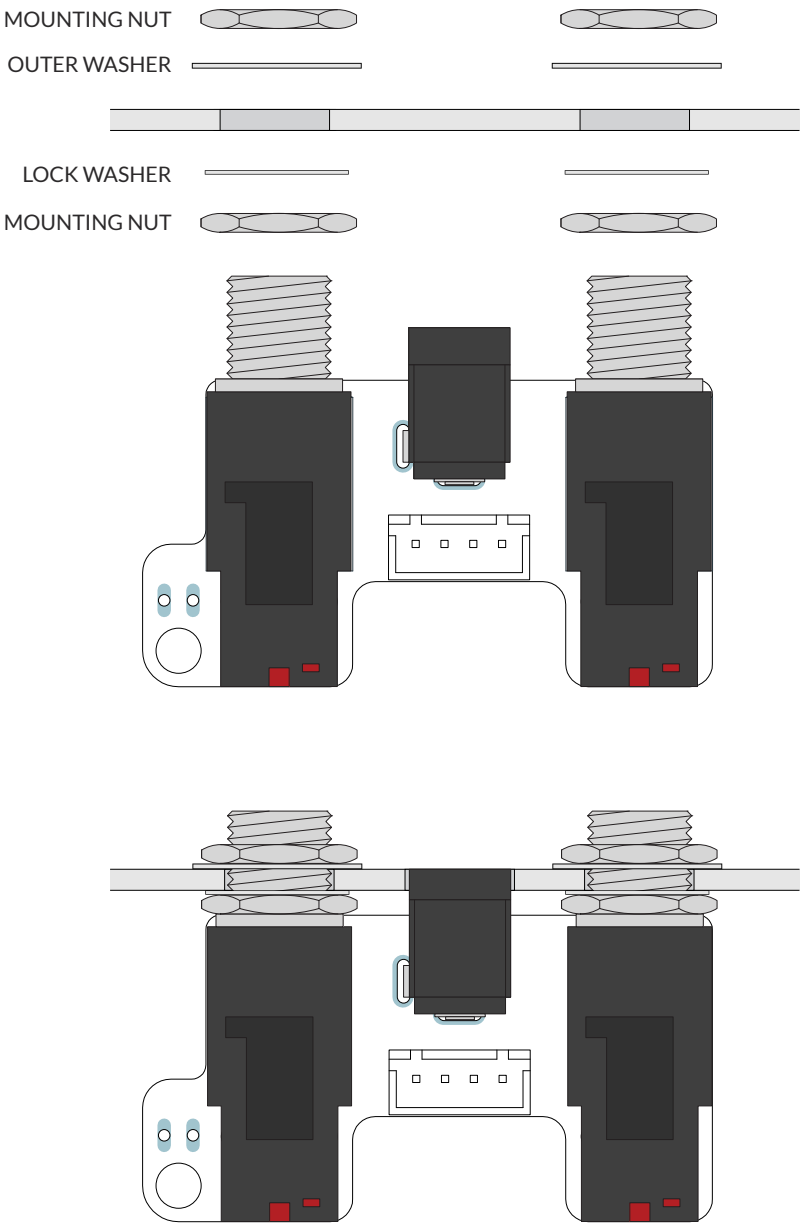
If you've read the documentation carefully and followed all the instructions, there's a good chance you will get it right the first time!

ENCLOSURE LAYOUT: INPUT/OUTPUT PCB

Affix the input/output PCB to the north-facing panel of the enclosure as shown.

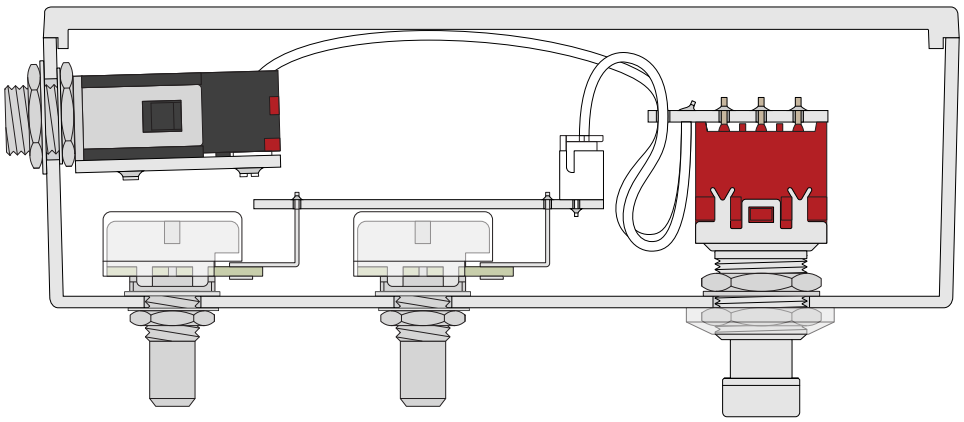
Note the use of two mounting nuts on each of the jacks, one inside and one outside. The inner nut acts as a spacer to set the DC jack flush with the outside of the enclosure. The inner nuts should be threaded as far down as they can go.





FINAL TESTING & ASSEMBLY

After everything is in place, plug the 3 wire assemblies into their respective headers and make sure they're secure. Here is a cross-section of the inside of the completed pedal.



At this point, you have completed the full circuit as far as the electrons are concerned. Plug in a 9-volt supply and test it out with a guitar and an amplifier.

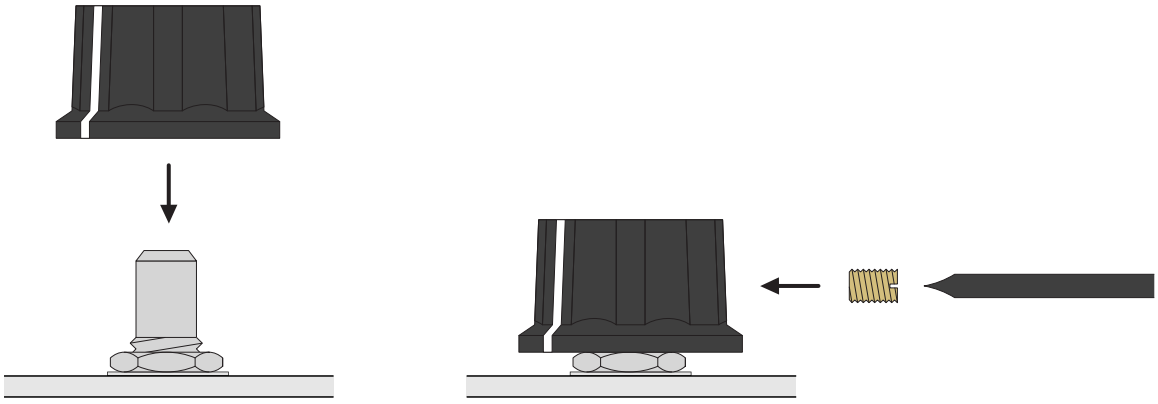
Test the bypass switch a few times, then start turning the knobs and see if everything sounds OK. If it works, great! If not, don't be discouraged. See page 27 for troubleshooting info.

Finishing touches

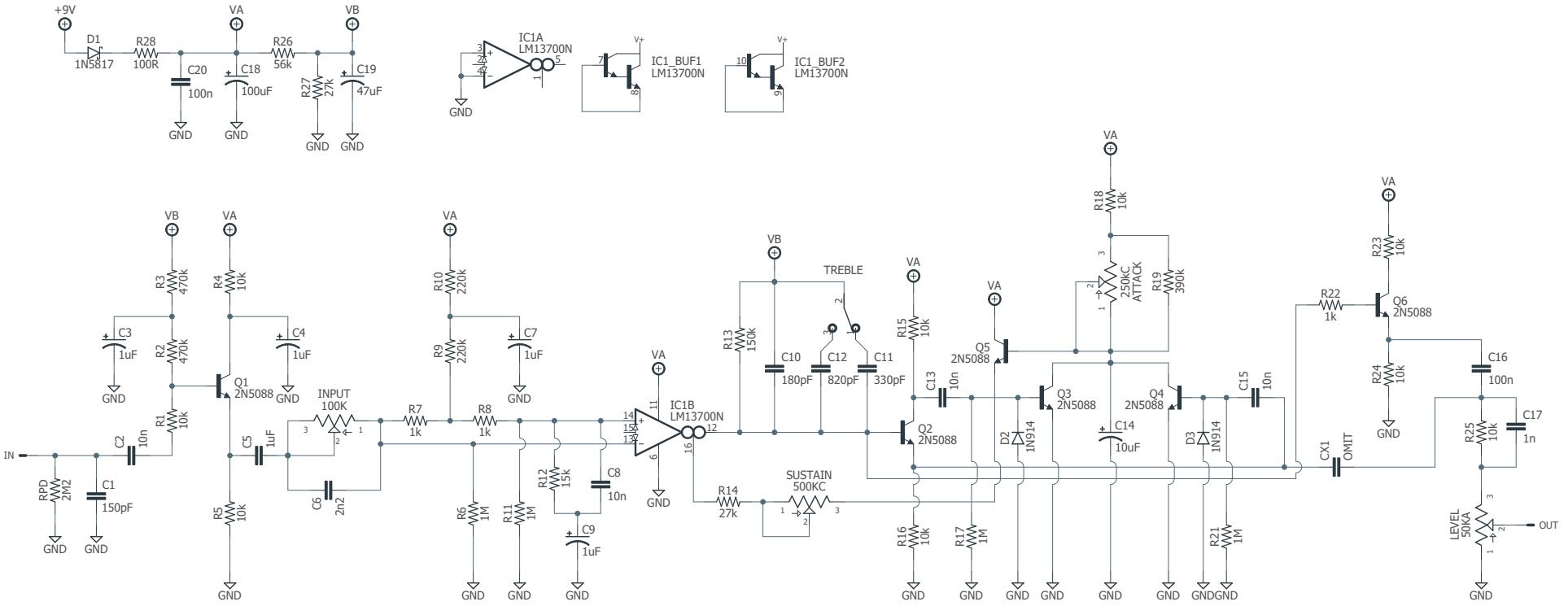
Now, just a couple of things for the final assembly. Turn the shafts all fully counter-clockwise, then put on the knob and rotate until the indicator line is aligned with the dot on the enclosure that shows the zero point. Affix the knobs to each of the potentiometer shafts as shown in the diagram below.

Using a small flat-head screwdriver (no more than 0.1" / 2.5mm in diameter), firmly tighten the set screw until it presses against the shaft of the potentiometer and holds the knob in place.

Be careful not to over-tighten or you may damage the set screw. But if it's not tight enough, the knob will be more likely to fall off or lose its alignment with the markings on the enclosure.



Last, just close the panel on the back using the four screws. That's it!


+9V
D1
1N5817
R28
100R
C20
100n
VA
C18
100uF
R26
56k
VB
C19
47uF
R27
27k
GND
IC1A
LM13700N
IC1_BUF1
LM13700N
IC1_BUF2
LM13700N
V+
R3
470k
R4
10k
R10
220k
R2
470k
R9
220k
C3
1uF
C4
1uF
C7
1uF
Q1
2N5088
R1
10k
IN
C2
10n
RPD
2M2
C1
150pF
C5
1uF
INPUT
100K
R7
1k
R8
1k
C6
2n2
R5
10k
R6
1M
R11
1M
R12
15k
C8
10n
C9
1uF
IC1B
LM13700N
R13
150k
TREBLE
C10
180pF
C12
820pF
C11
330pF
R14
27k
SUSTAIN
500KC
R15
10k
R16
10k
R17
1M
C13
10n
Q2
2N5088
Q5
2N5088
R18
10k
250kC
ATTACK
R19
390k
Q3
2N5088
Q4
2N5088
D2
1N914
D3
1N914
C14
10uF
C15
10n
CX1
OMIT
R21
1M
R22
1k
R23
10k
Q6
2N5088
R24
10k
C16
100n
C17
1n
R25
10k
LEVEL
50KA
OUT

FULL PARTS LIST

Resistors

PART	VALUE
R1	10k
R2	470k
R3	470k
R4	10k
R5	10k
R6	1M
R7	1k
R8	1k

PART	VALUE
R9	220k
R10	220k
R11	1M
R12	15k
R13	150k
R14	27k
R15	10k
R16	10k

PART	VALUE
R17	1M
R18	10k
R19	390k
R21	1M
R22	1k
R23	10k
R24	10k
R25	10k

PART	VALUE
R26	56k
R27	27k
R28	100R
RPD	2M2
LEDR	10k

Capacitors

PART	VALUE
C1	150pF MLCC
C2	10n film
C3	1uF electro
C4	1uF electro
C5	1uF film
C6	2n2 film
C7	1uF electro

PART	VALUE
C8	10n film
C9	1uF electro
C10	180pF MLCC
C11	330pF MLCC
C12	820pF MLCC
C13	10n film
C14	10uF

PART	VALUE
C15	10n film
C16	100n film
C17	1n film
C18	100uF electro
C19	47uF electro
C20	100n MLCC
CX1	OMIT

Diodes

PART	VALUE
D1	1N5817
D2	1N914
D3	1N914

IC & Sockets

PART	VALUE
IC1	LM13700 or NE5517
IC1SA	DIP-8 socket
IC1SB	DIP-8 socket

Transistors

PART	VALUE
Q1	2N5088
Q2	2N5088
Q3	2N5088
Q4	2N5088
Q5	2N5088
Q6	2N5088

Potentiometers

PART	VALUE
Sustain	500kC
Attack	250kC
Volume	50kA

Attack is labeled "Release" on the enclosure.

Switches

PART
SPDT on-off-on
3PDT stomp

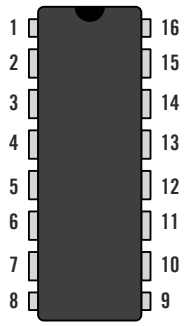
TROUBLESHOOTING INFORMATION

If you finish building the kit and find that it doesn't work right, we've written a separate in-depth Troubleshooting Guide that applies to all of our kits. The main troubleshooting process is covered there. Here you will find information specific to this kit that will help with that process.

Voltages

The following voltages are taken from our prototype unit using a **9.60V** supply. Your measured voltages won't be exactly the same due to variance in power supplies and component tolerances. However, if you see anything more than +/-0.5V from the listed voltages, it's a good indicator of an issue, and the exact voltages can help narrow it down.

Note that IC pins are labeled counter-clockwise from the upper-left, as shown in the diagram to the right. For JFETs, the legs will be marked on the PCB.



IC1

PIN	VOLTAGE
3	0
4	0
6	0
11	9.30
12	2.42
13	4.63 (falling)
14	4.62 (falling)
15	0.61
16	1.33

Q1

PIN	VOLTAGE
E	1.95
B	2.42
C	7.24

Q2

PIN	VOLTAGE
E	1.88
B	2.42
C	7.33

Q3

PIN	VOLTAGE
E	0
B	0
C	9.05

Q4

PIN	VOLTAGE
E	0
B	0
C	9.05

Q5

PIN	VOLTAGE
E	8.50
B	9.05
C	9.21

Q6

PIN	VOLTAGE
E	1.88
B	2.42
C	7.33

Note: The pins not listed for IC1 do not have any electrical connection to the rest of the circuit and do not need to be tested. Measurements were taken with Sustain knob all the way up and Attack all the way down, which affects the voltage slightly.

SUPPORT

Aion FX does not offer direct support for these projects beyond the provided documentation. Replacements and refunds cannot be offered unless it can be shown that the circuit or documentation are in error or that the included components are non-functional.

Where to get help

The three best places to ask for help are the DIY Stompboxes forum, the DIY Stompboxes Facebook group, and the r/diypedals subreddit. These communities have more than 150,000 members between them and they are very accommodating to new builders.

When posting a troubleshooting request, always include the following:

1. A thorough description of the problem you are experiencing
2. A photo of the inside of the pedal
3. A list of all the measured voltages of each of the pins, described on the previous page

While we cannot offer direct, private support, you may send a link to your public troubleshooting thread to Aion FX using the contact form on the website. There is no guarantee that we will be able to join the discussion and help solve your problem, but this improves the chances.

It benefits the whole community if the troubleshooting process is public because then people who have the same issue in the future may come across it when searching. And if you do get help, remember to pay it forward! The best way to learn new skills is to help others. Even if you've only built one pedal, you have more experience than someone who is brand new, so you have something to offer.

RESALE TERMS

These kits may be used for commercial endeavors in any quantity unless otherwise noted. It's okay to sell individual builds locally or online, or even to offer a service to build pedals based on these kits.

No direct attribution is necessary, though a link back is always greatly appreciated. The only usage restriction is that you cannot "goop" the PCB or otherwise obscure the source. In other words: you don't have to go out of your way to advertise the fact that you use Aion FX kits, but please don't go out of your way to hide it. The guitar effects industry needs more transparency, not less!

LEGAL INFORMATION

MXR® and Dyna Comp® are registered trademarks of Dunlop Manufacturing, Inc.

All other trademarks are property of their respective owners.

Any use of trademarks is for comparative advertising purposes only under fair use. It is not an endorsement of this product by the trademark holders.

These kits are intended to be built by the customer. Aion FX is not responsible for language that may be used by the customer in the marketing or resale of the finished product.

DOCUMENT REVISIONS

1.0.1 (2024-08-08)
Added link to troubleshooting guide on page 27.

1.0.0 (2022-11-11)
Initial release.